Using ABAQUS Scripting Interface for Materials Evaluation and Life Prediction

Lynn M. Powers

Ohio Aerospace Institute, NASA Glenn Research Center, Cleveland, OH 44135

Steven M. Arnold

NASA Glenn Research Center, Cleveland, OH 44135

Andrzej Baranski

ABAQUS, Inc., Providence, RI, 02909

Abstract: An ABAQUS script has been written to aid in the evaluation of the mechanical behavior of viscoplastic materials. The purposes of the script are to: handle complex load histories; control load/displacement with alternate stopping criteria; predict failure and life; and verify constitutive models. Material models from the ABAQUS library may be used or the UMAT routine may specify mechanical behavior. User subroutines implemented include: UMAT for the constitutive model; UEXTERNALDB for file manipulation; DISP for boundary conditions; and URDFIL for results processing. Examples presented include load, strain and displacement control tests on a single element model. The tests are creep with a life limiting strain criterion, strain control with a stress limiting cycle and a complex interrupted cyclic relaxation test. The techniques implemented in this paper enable complex load conditions to be solved efficiently with ABAQUS.

Keywords: Experimental Verification, Scripting, Deformation, Creep, Fatigue, Finite Element.

1. Introduction

A prerequisite for meaningful assessment of component durability and life, and consequently design of structural components, is the ability to accurately predict stresses, strains, failure modes and their subsequent interaction and evolution occurring within a loaded structure. Since constitutive material models provide the required link between stress and strain, this necessitates the development, characterization and validation of an appropriate constitutive behavior model for any material. Unified, internal state variable, models enable one to capture interaction effects like

plasticity and creep (or relaxation) whereas classical non-unified approaches miss these interaction effects which can significantly impact the ability to predict the deformation response and life of components. The generalized viscoelastoplastic with potential structure (**GVIPS**) class constitutive model, developed by Arnold and Saleeb (1994) and extended to included a wide range of relaxation spectrums by incorporating multiple mechanisms (Saleeb, et al, 2001, Saleeb and Arnold, 2004), represents such a unified model. The utilization of this model within the ABAQUS finite element analysis program has been previously accomplished by implementing this model within ABAQUS's user material subroutine (UMAT). An ABAQUS script has been written to assist constitutive model developers and analysts to perform two primary tasks associated with understanding and validating the performance of a given model. The first task is to efficiently conduct simulations involving complex load histories where control variables (e.g. stress or strain), i.e., modes of loading, and target variables (e.g. stress, strain, time, displacement, etc) are present. The second is related to the first, in that it provides the ability to perform a variety of relaxation trajectories given various starting locations within the state space (see Onat and Fardshisheh, 1973; Arnold, 1990 for details) and graphically display these trajectories. This "state space" representation is often extremely helpful in understanding theoretical and numerical performance of a given model. Note this script is executed from ABAQUS/CAE and any constitutive model, be it within ABAQUS or one specified within a given UMAT routine can be utilized with the present script.

The experiments described and utilized herein were conducted by Lissenden and coworkers (Lissenden, et al, 2005) to help characterize the macroscopic time-dependent deformation of a titanium alloy (i.e., TIMETAL 21S). This data was used to validate the capability of the recent multi-mechanism representation of a unified viscoelastoplastic model (Arnold, et al, 2005). The examples presented correspond to some of these response curves which include: a life limiting creep test, a strain control cyclic test with stress limits, and a complex strain control cyclic test with interrupted relaxation tests.

2. ABAQUS Script

The script directs the process of analyzing a complex laboratory or field test in which the control variable and the target variable are not required to be of the same type. This is accomplished by executing a single Python script within the ABAQUS/CAE environment wherein the output file is continually monitored so as to obtain the material response to the imposed history. In this way variables that are not being specified (or controlled) can be used to end a given "load" step. The following steps are executed:

1. Delete files from a previous analysis in this directory
2. Create geometry, a 1x1x1 cube with surfaces and node sets defined
3. Import material properties for the UMAT
4. Generate initial step to setup the analysis
5. Read load control history
6. For each load step
    - assign load control and targets
    - generate ABAQUS input file

- restart ABAQUS while monitoring target values in the jobname.fil file

7. Process results and generate reports for all load steps

The first step cleans the current directory for a new analysis. It is recommended that a new folder is used. The geometry is created with surfaces defined for load application in any direction.

The material constants are input from a file named "user_constants.txt". The number and order of these constants are dependent on the UMAT configuration. Any combination of several load conditions may be specified within the script; these include:

1. Constant strain rate
2. Constant stress rate
3. Creep, i.e., constant load
4. Relaxation, i.e., constant strain
5. Cyclic, load/unload

For example, for a creep or relaxation test, an initial load up step is required at the beginning of the test. Now, with the availability of this new script it is possible to conveniently use either a constant stress rate or strain rate step since the loading variable and target variable are decoupled. This is demonstrated in the example section. Note, each load step is terminated when a target condition reaches a target value. If the history continues, a new loading condition is applied. Current target variables include: 1) time, 2) stress, 3) strain, or 4) displacement.

3. Basis of Strain-Control Algorithm

For illustration purposes an outline of the basis for conducting strain-controlled loadings will be briefly described. In the current version of ABAQUS, there is no functionality to prescribe a strain control load condition. However, a strain controlled load can be modeled using a displacement control function which is updated after each time increment to enforce strain control or a constant strain rate. The displacement control function will be modeled using the DISP subroutine in the following way. The logarithmic strain, ε , is:

$$\varepsilon = \ln\left(\frac{l}{l_0}\right) = \ln\left(\frac{l_0 + u}{l_0}\right)$$

where l_0 is the original length, l is the current length and u is the displacement. The strain increment is:

$$\varepsilon_{n+1} = \dot{\varepsilon}_{n+1}\Delta t + \varepsilon_n$$

where $\dot{\varepsilon}$ is the strain rate. The increment n is the last converged time step, $n+1$ is the next time step and Δt is the duration of the $n+1$ time step. The prescribed displacement is:

$$u_{n+1} = l_0\left[\exp\left(\dot{\varepsilon}_{n+1}\Delta t + \varepsilon_n\right) - 1\right]$$

Since the logarithmic strain at the previous time step is not passed to the user subroutine DISP, the user subroutine URDFIL is implemented to pass this information through a common block.

4. Demonstration Examples

Four examples are shown to demonstrate the utility and output of the present script; these are 1) multistep stress controlled test, 2) a creep test with a life limiting strain criterion, 3) a strain control, stress limited cyclic test, and 4) two interrupted cyclic/relaxation tests. Note examples two through four demonstrate realistic histories actually performed in the laboratory which could not be precisely analyzed using ABAQUS without the current script. Furthermore, these examples provide evidence of the predictive capability of the GVIPS model. The material tested was a titanium alloy (TIMETAL 21S) . The titanium alloy shows both reversible and irreversible behavior. The characterization of this material was based on constant strain rate, creep and relaxation tests (Arnold, et al, 2005). The material parameters were determined with COMPARE (COnstitutive Material PARameter Estimator) and can be found in Saleeb, et al, 2004.

4.1 Multistep Stress Controlled Test

Table 1 specifies the control and target variables for the multistep stress control test. The steps include: a load up; creep to a specified strain; unload to a specified stress; creep for a specified time; and reload. Time limits on the step duration are necessary to terminate the analysis if the target value is not reached.

Figure 1 shows the plots generated by the ABAQUS script. Each of these is generated in its own viewport. Figure 1a shows the load history or the stress as a function of time. Figure 1b shows the stress-strain curve for the test. Logarithmic strain as a function of time is shown in Figures 1c. The final plot (Figure 1d) shows the stress as a function of an internal variable. This state space plot is dependent on user input and which internal variables need to be displayed.

In addition to plots generated by the script, report files are also generated. These are useful to export ABAQUS history data for comparison with experimental results or further processing with other tests, etc. The report feature was used in all examples.

Table 1. History specification for multistep stress controlled test.

Step	Control Variable	Change in load/control parameter	Max. Step Duration (seconds)	Target Variable	Target Value
1	S11	40 ksi (280 MPa)	40	S11	40 ksi (280 MPa)
2	S11	0	720	LE11	0.025
3	S11	-60 ksi (-420 MPa)	60	S11	-20 ksi (-140 MPa)
4	S11	0	720	Time	720 sec
5	S11	50 ksi (350 MPa)	50	Time	50 sec

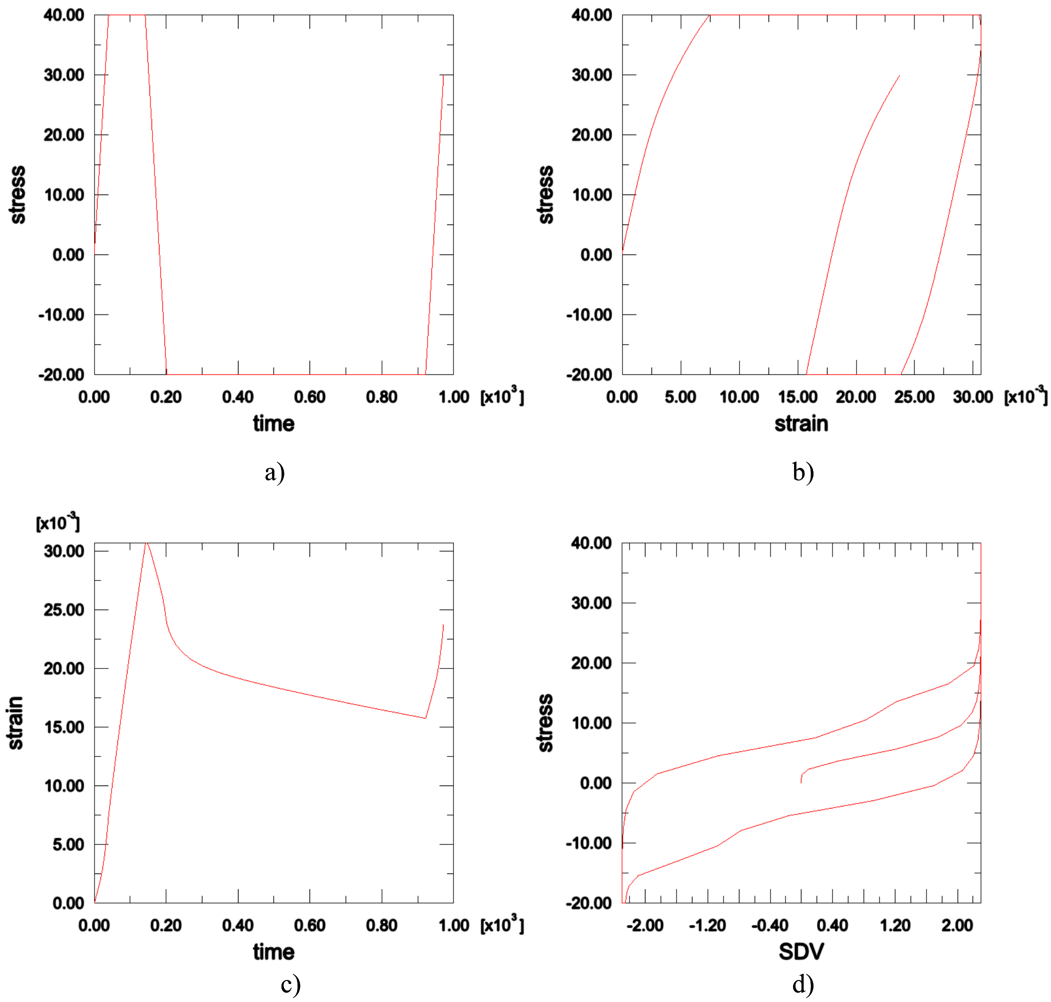

Figure 1. Plots generated during the script execution for the load history given in Table 1, a) stress-time, b) stress-strain, c) strain-time, d) state space, stress as a function of internal variable. Units are ksi for stress, seconds for time.

4.2 Creep test

The time needed to reach a strain level in a creep test is difficult to determine. Using this script, the strain limit may be specified and the analysis will continue until that limit is reached. For the one element model used for this example the additional analysis time is trivial. However, for a large model the time savings could be substantial.

The load history with targets for a 128 MPa (18.6 ksi) creep test is given in Table 2. The first step is the load up and the second is the constant load condition. The target strain for this demonstration is 0.03. The control condition is prescribed in one direction, whereas the target variable may be in another direction.

Figure 2 shows the strain as a function of time for the creep test. The ABAQUS results show that a time of 3.25 hours is estimated to reach a strain of 0.03.

Table 2. History specification for a creep test.

Step	Control Variable	Change in load/control parameter	Step Duration (seconds)	Target Variable	Target Value
1	S11	128 MPa	1.86	Time	1.86 sec
2	S11	0	18,000 (5 hr)	E11	0.03

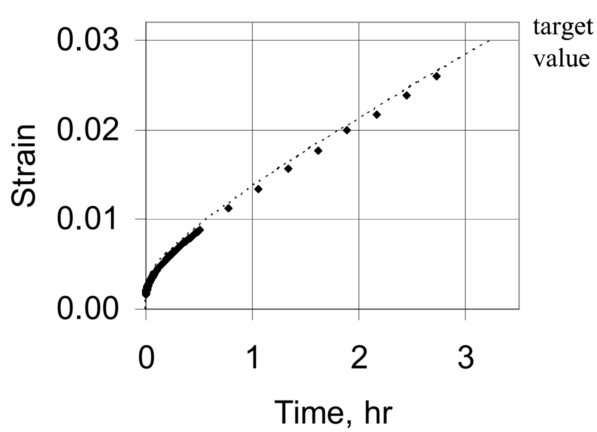

Figure 2. Strain as a function of time for a 128 MPa creep test. The diamonds are the experimental data and the dashed line is the ABAQUS result.

4.3 Strain-Controlled, Stress Limited Cyclic Test

The power of the ABAQUS script is shown in a multistep test with different load control and target variables. The cyclic test consists of strain control loading with a stress limit as a target criterion. For this example the applied strain rate is 0.001 as illustrated in Table 3. A series of 30 cycles are analyzed. The load history is shown in Table 3.

Figure 3 shows the resulting stress strain curve for the imposed history. The analysis would be difficult without the target criterion, as the duration of each step is dependent on the stress target.

Table 3. History specification for strain-controlled, stress limited cyclic test.

Step	Control Variable	Change in load/control parameter	Step Duration (seconds)	Target Variable	Target Value
1	LE11	0.005	5	S11	280 MPa (40 ksi)
2	LE11	-0.01	10	S11	-280 MPa (-40 ksi)
3	LE11	0.01	10	S11	280 MPa (40 ksi)
…					
59	LE11	0.01	10	S11	280 MPa (40 ksi)
60	LE11	-0.01	10	S11	-280 MPa (-40 ksi)

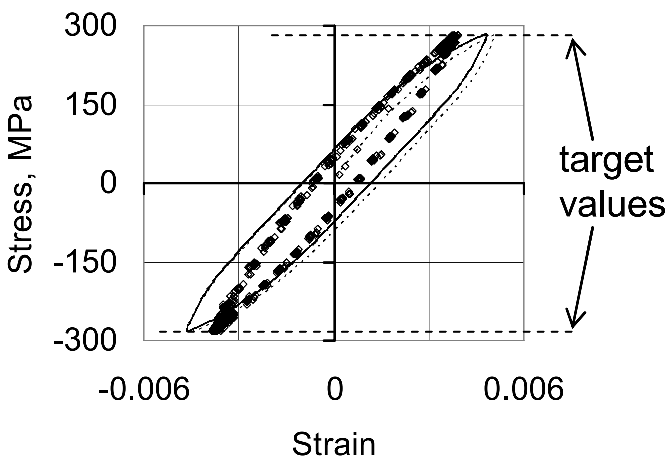

Figure 3. Stress Strain curve for the cyclic test. Experimental data are represented by the symbols and the ABAQUS results by the dashed line.

4.4 Interrupted Cyclic Relaxation Tests

These two final examples demonstrate the utility of the present script given complex loading conditions. Two tests, labeled 87-50 and 87-21 respectively, were conducted to study the path dependent behavior of the titanium alloy, TIMETAL 21S. In total, each test consisted of a cycle with multiple two hour relaxation histories within the cycle followed by approximately 50 strain controlled cycles. The test is repeated 3 times in tension and than three times in compression. The goal of this study was to determine if the time dependent material behavior exhibited was significantly impacted by the prior history, i.e., path dependent. This was demonstrated with this study.

The load history specified in the ABAQUS input file for each test is given explicitly in Table 4, with the corresponding strain-time histories for both tests 87-50 and 87-21 being shown in Figures 4a and 4b respectively. Both tests are strain controlled, however test 87-50 uses only time as target variable whereas test 87-21 involves both time and stress as target variables. Further test 87-50 contains eight two hour relaxation histories within each cycle whereas test 87-21 only contains three 2 hour relaxation histories within each cycle since test 87-21 was directly unloaded to the same stress level that was obtain from test 87-50 at a strain level of 0.003 subsequent to unloading and the first relaxation history. The corresponding response (stress-strain) histories are shown in Figures 5a and 5b, respectively. Note the difference in strain magnitude, due to prior history, even though the stress levels are the same between histories.

Table 4. History specification of cyclic relaxation tests.

		87-50				87-21			
Step	Control Variable	Change in load/control parameter	Step Duration (seconds)	Target Variable	Target Value	Change in load/control parameter	Step Duration (seconds)	Target Variable	Target Value
1	LE11	0.005	5	Time	5	0.005	5	Time	5
2	LE11	-0.001	1	Time	1	-0.004	4	S11	-59.4 MPa
3	LE11	0	7200	Time	7200	0	7200	Time	7200
4	LE11	-0.001	1	Time	1	-0.003	3	S11	-153.0 MPa
5	LE11	0	7200	Time	7200	0	7200	Time	7200
6	LE11	-0.002	2	Time	2	-0.003	3	S11	-177.4 MPa
7	LE11	0	7200	Time	7200	0	7200	Time	7200
8	LE11	-0.002	2	Time	2	0.01	10	Time	10
9	LE11	0	7200	Time	7200				
10	LE11	-0.003	3	Time	3				
11	LE11	0	7200	Time	7200				
12	LE11	-0.001	1	Time	1				
13	LE11	0.001	1	Time	1				
14	LE11	0	7200	Time	7200				
15	LE11	0.001	1	Time	1				
16	LE11	0	7200	Time	7200				
17	LE11	0.002	2	Time	2				
18	LE11	0	7200	Time	7200				
19	LE11	0.001	1	Time	1				

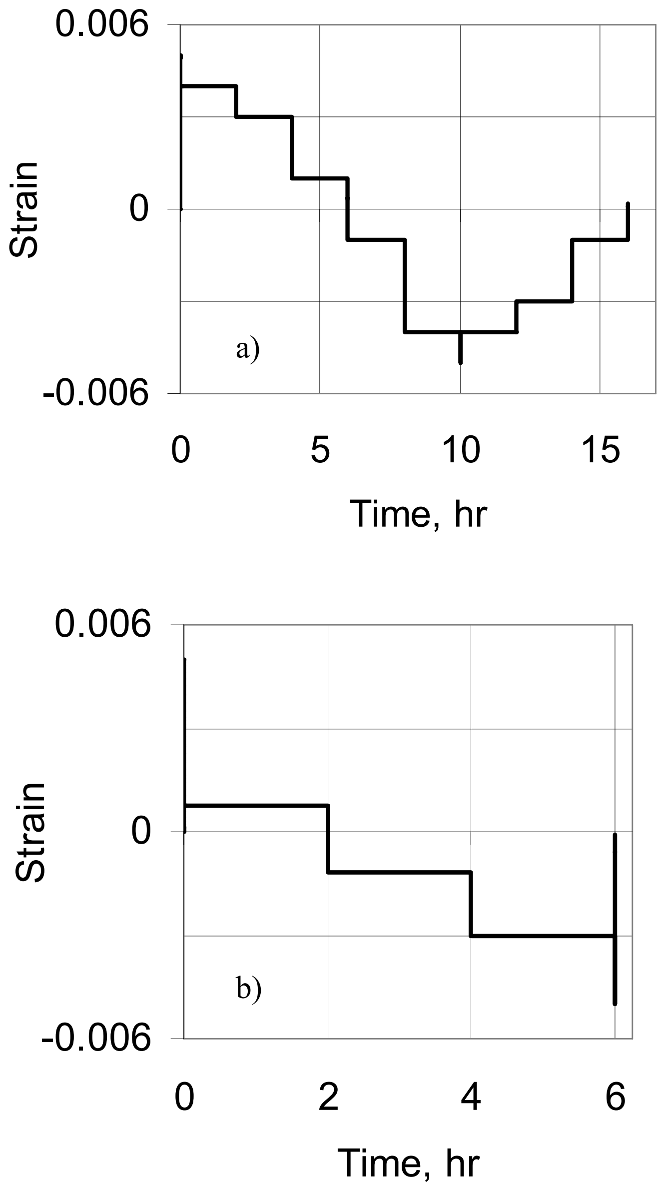

Figure 4. Strain for the cyclic relaxation test. The first cycle is shown for a) test 87-50 and b) 87-21.

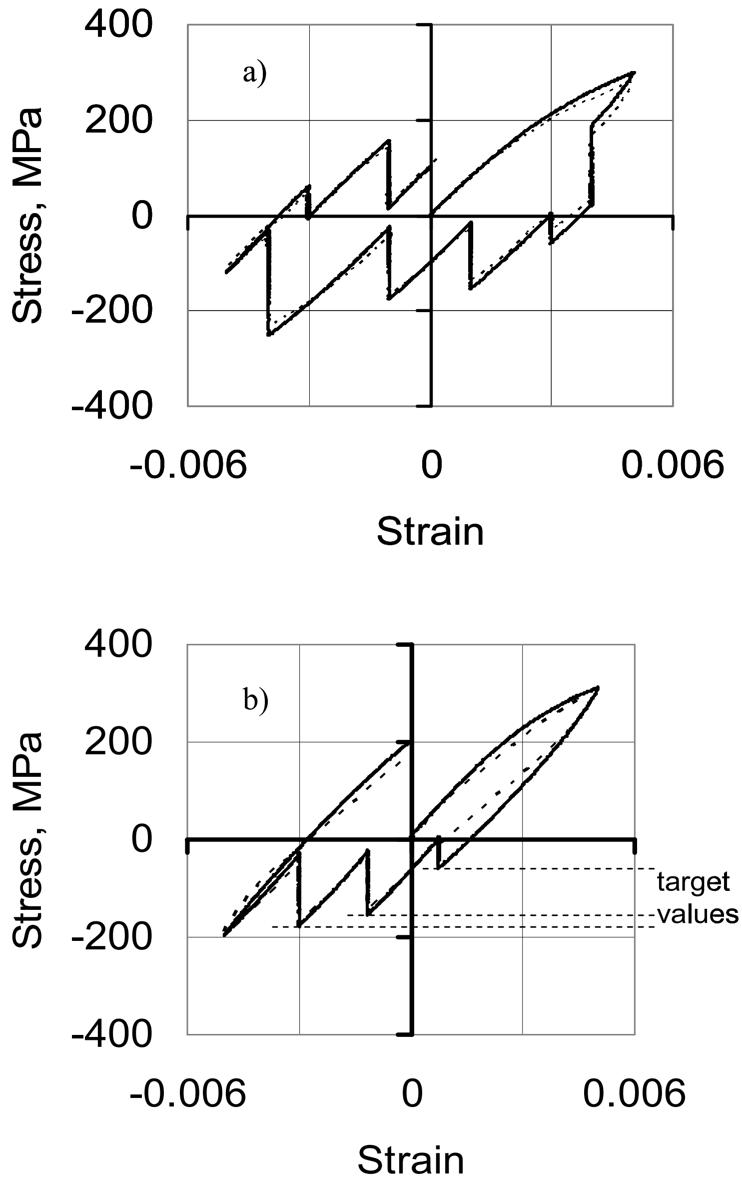

Figure 5. Stress strain curve for titanium with relaxation for a) test 87-50 and b) 87-21. Solid line is experiment and dashed line is ABAQUS.

5. Concluding Remarks

Complex load conditions may be prescribed efficiently with a new ABAQUS script for a one element model. The power of this script is demonstrated with examples which have different load and target conditions. Finally, plots and reports are generated to summarize the results. ABAQUS scripts may be written to handle complex load strategies for any model geometry.

Future work will involve enhancing this methodology to accommodate component level analysis. A modified script will be written to apply loads to section(s) of the model with target variables in other sections. Obviously, the mesh geometry would need to be customized for each specific analysis. Another piece of the script would be dedicated to load control options throughout the model, with target conditions being specified at one or more locations.

6. References

1. Arnold, S. M**.**, ‘‘Quantification of Numerical Stiffness for a Unified Viscoplastic Constitutive Model”, Journal of Engineering Materials and Technology, Vol. 112, pp. 271-276, 1990.
2. Arnold, S. M., and Saleeb, A.F., ‘‘On the Thermodynamic Framework of Generalized Coupled Thermoelastic-Viscoplastic -Damage Modeling”, International Journal of Plasticity, Vol. 10, No. 3, pp. 263-278, 1994.
3. Arnold, S.M., Saleeb, A.F., Powers, L. and Lissenden, C.J., "High Temperature Characterization and Prediction of Timetal 21S Cyclic and Cyclic-Relaxation Deformation Behavior Using a Multimechanism Viscoelastoplastic Model" Plasticity 2005, Hawaii, Jan 4 - 8, 2005.
4. Lissenden, C.J., Doraiswamy, D., Solimine, P.A., and Arnold, S.M., "Experimental Investigation of Cyclic and Time-Dependent Deformation of Titanium Alloy at Elevated Temperature", Plasticity 2005, Hawaii, Jan 4 - 8, 2005; Proceedings: Dislocations, Plasticity, Damage and Metal Forming: Material Response and Multiscale Modeling, A.S. Khan and A.R. Khoei, Eds., Neat Press, pp. 217-219, 2005.
5. Onat, E.T. and Fardshisheh, F., “Representation of Creep, Rate Sensitivity and Plasticity, SIAM, Jnl of Applied Math., Vol. 25, No. 3, pp. 522-538, 1973.
6. Saleeb, A.F., Arnold, S.M., Castelli, M.G , Wilt, T.E., and Graf, W.E., “A General Hereditary Multimechanism-Based Deformation Model with Application to The Viscoelasto-plastic Response of Titanium Alloys”, International Journal of Plasticity, Vol. 17, No. 10, pp. 1305-1350, 2001.
7. Saleeb, A.F. and Arnold, S.M., “Specific Hardening Function Definition and Characterization of A Multimechanism Generalized Potential-Based Viscoelasto-plasticity Model”, International Journal of Plasticity, Vol. 20, pp. 2111-2142, 2004.
8. Saleeb, A.F. Marks, J.R., Wilt, T.E. and Arnold, S.M., “Interactive Software for Material Parameter Characterization of Advanced Engineering Constitutive Models”, Adv. Eng. Software, Vol. 35, pp. 383-398, 2004.